

## INLETS, DUCTS, AND NOZZLES

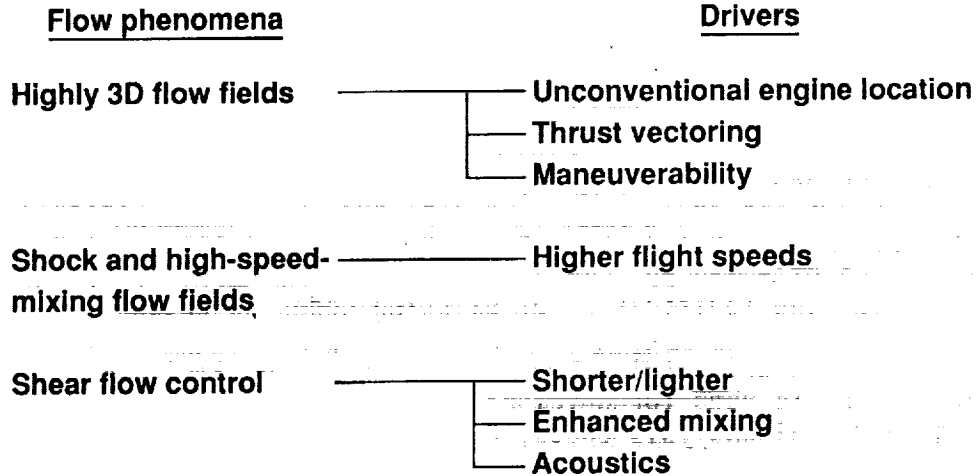
James R. Scott and John M. Abbott  
NASA Lewis Research Center  
Cleveland, Ohio

The internal fluid mechanics research program in inlets, ducts, and nozzles is a balanced effort between the development and application of computational tools and the conduct of experimental research. The computational effort involves the development and validation of advanced computational fluid dynamics (CFD) codes through comparison with data, the modification of existing codes to extend their range and accuracy, and the application of codes to practical problems to demonstrate their value in design. The experimental research involves both simplified and realistic complex geometries and is used for developing flow physics understanding, for validating advanced numerical analysis codes, and for developing physical models of flow phenomena.

The inlet, duct, and nozzle research program is described according to three major classifications of flow phenomena: (1) highly three-dimensional flow fields, (2) shock and high-speed-mixing flow fields, and (3) shear flow control. Specific examples of current and future elements of the research program are described for each of these phenomena. In particular, the highly three-dimensional flow field phenomenon is highlighted by describing the experimental and computational research program in transition ducts having a round-to-rectangular area variation. In the case of shock and high-speed-mixing flow fields, both experimental and computational results are presented for the mixing of a high-speed stream injected into a second high-speed stream. For shear flow control, research in the use of aerodynamic excitation to enhance the jet mixing process is described. In addition, results that stem from using small tabs protruding into a nozzle exit flow stream to enhance mixing are also presented.

The presentation is concluded by describing a three-dimensional, unsteady viscous code development effort that will provide a well-documented, user-friendly flow solver for computing inlet, duct, and nozzle flow fields in the future.

## Inlets, Ducts, and Nozzles Research Program



CD-91-54092

The internal fluid mechanics research program in inlets, ducts, and nozzles is described according to three types of fluid flow phenomena - highly three-dimensional flow fields, shock and high-speed-mixing flow fields, and shear flow control. The importance of each of these flow phenomena is a result of the drivers listed in the figure. For example, highly three-dimensional internal flow fields result from unconventional engine locations where twisting and turning inlets, ducts, and nozzles must be designed to deliver the airflow to and from the free stream. Aircraft thrust vectoring requirements quite often lead to the transitioning of nozzle cross-sectional geometries from round to rectangular with resultant three-dimensional flows. Aircraft maneuverability requirements can lead to strong three-dimensional flow fields entering the propulsion system inlet and ducting system. The push toward higher flight speeds, for both military and civilian applications, leads to the importance of research in shock and high-speed-mixing flows, particularly in terms of interactions (e.g., shock/boundary layer interactions). The desire to design inlets, ducts, and nozzles to be as short and light as possible points to the importance of shear flow control as a means of "stretching" the limits of the geometry while avoiding internal flow separations. Shear layer control in another sense (i.e., the use of aerodynamic excitation to control the formation and development of a mixing layer) offers the potential for enhancing the mixing process in external nozzle flows.

## **Inlets, Ducts, and Nozzles Research Program**

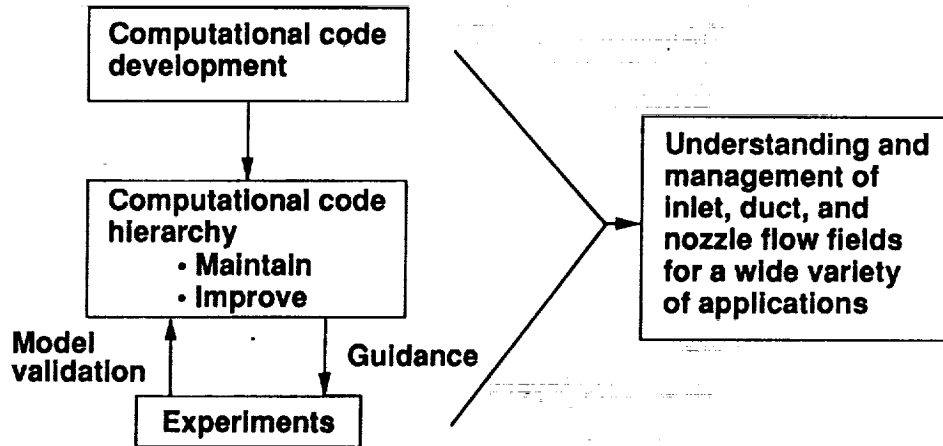
### **Flow Phenomena**

- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**
- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

CD-91-54093

Specific elements of the inlet, duct, and nozzle research program are listed for each of the three flow field phenomena. The three elements highlighted in the figure will be expanded upon in the remainder of the presentation. Specifically, highly three-dimensional flow fields will be illustrated by describing the transition duct research program. The shock and high-speed-mixing flow field phenomena will be illustrated with a description of the high-speed-mixing research program. Shear layer control research will be illustrated with examples from the free-shear-layer control element of the program. The remaining three major elements of the overall program (i.e., offset ducts, shock/boundary layer interactions, and boundary layer control) will not be described in this presentation, although they are also key elements of the overall program.

## Inlets, Ducts, and Nozzles Research Program Approach



CD-91-54094

The approach to the research program in inlets, ducts, and nozzles is balanced between computational and experimental research. As shown in the figure, the ultimate goal of the program is to develop understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications. This is accomplished through an interactive process in which a strong computational effort of developing, maintaining, and improving CFD codes works hand in hand with an experimental effort that provides the data necessary for code validation and flow phenomena modeling. The codes within the hierarchy, in turn, provide guidance for conducting the experiments in terms of where measurements should be concentrated and what values of test parameters should be selected.

## **Inlets, Ducts, and Nozzles Research Program**

### **Flow Phenomena**

- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**
- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

CD-91-54095

Highly three-dimensional flow field research will be illustrated by describing the transition duct research program.

## Transition Duct Research

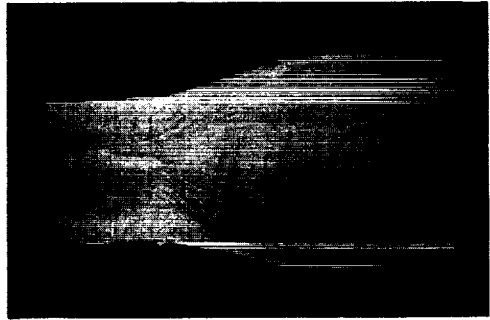


CD-91-54096

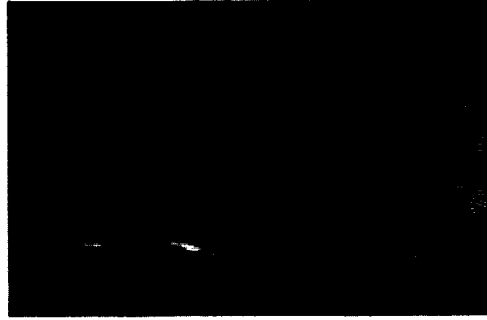
Transition ducts are characterized by a transition in cross-sectional shape from round to rectangular with downstream distance. Shown in the figure is a transition duct installed in the Internal Fluid Mechanics Test Facility. This particular duct geometry transitions from a round cross section to a rectangular cross section having an aspect ratio (width/height) of 3.0. The duct is three inlet diameters long and has an exit-to-inlet area ratio of 1.0. In the design of the test facility, special care was taken to condition the transition duct inflow properly to provide the desired levels of inflow turbulence, flow angularity, and boundary layer profile. In addition, a swirl generator was designed and built to provide a solid-body-rotation swirling flow into the transition duct for evaluation of how swirl affects duct performance. Swirl can become an important consideration for transitioning nozzle applications where residual swirl from the engine turbine may be convected into the transition duct. The facility is capable of being operated either with an atmospheric inlet (as shown here) or connected to a high-pressure supply system to provide higher Reynolds number test capability.

## Transition Duct Flow Visualization Results

Axial Mach Number= 0.35



Surface oil streaks  
(nonswirling inflow)

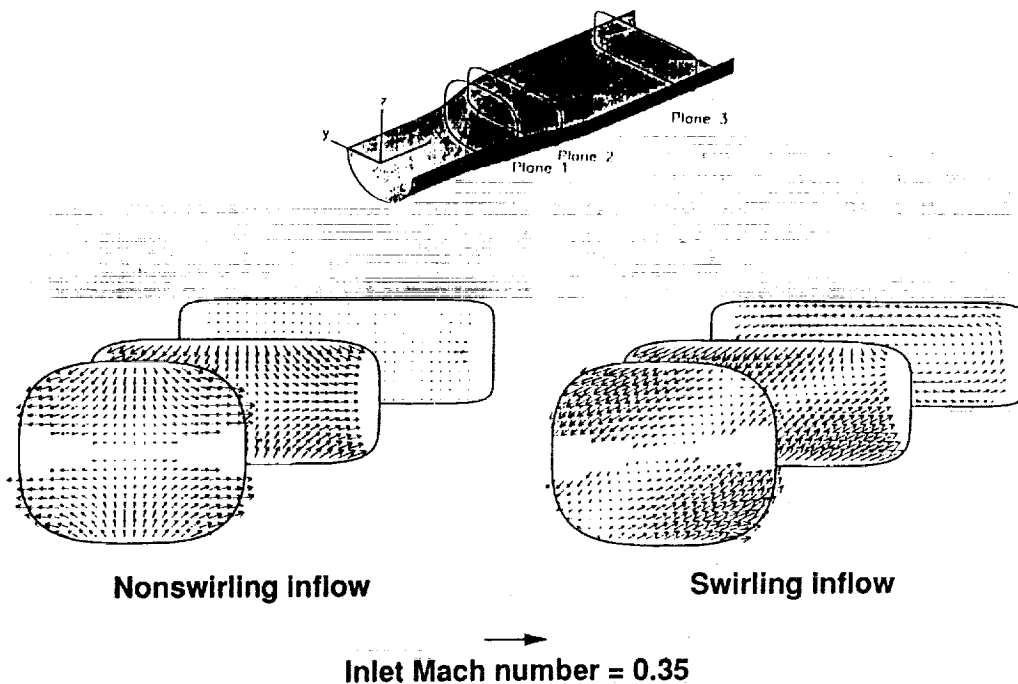


Surface oil streaks  
(swirling inflow)

CD-91-54097

Surface oil flow visualization results are shown in this figure for both the non-swirling and the swirling inflow cases with an inflow Mach number of 0.35. The flow is from left to right and indicates that for both cases the surface flow remained attached over the full length of the duct. The swirling inflow is clearly evident for the swirl case. The technique used to obtain these results consists of applying a uniform layer of oil containing a fluorescent dye to the surface of the duct and then operating the facility at the appropriate test condition for several minutes to let the oil flow and form streaks in the direction of the flow field near the surface. The duct is then disassembled and the resulting oil flow pattern is photographed using ultraviolet light for illumination.

## Transition Duct Aerodynamic Results Secondary Mach Number Vectors



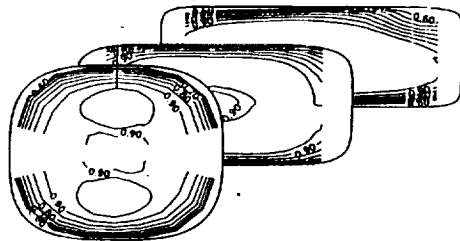
CD-91-54098

Experimental results are shown in this figure in terms of secondary Mach number vectors at three axial stations within the transition duct for both the nonswirling and the swirling cases. The duct inlet axial Mach number is 0.35, and for the swirl case the character of the swirl is close to that of solid-body rotation with an inlet swirl angle of about  $16^\circ$  at the duct wall. The "blank" regions within the duct represent areas where measurements could not be taken with the traversing probes. The values of Mach number were computed from measurements made with a calibrated five-hole probe. At each axial station, measurements were made at approximately 480 locations in the duct cross-sectional area. The results for the nonswirling inflow case show the development of a vortical flow structure at the duct exit plane as the flow has been influenced by the static pressure gradients associated with the duct area changes and curvatures in both the vertical and horizontal directions. Note also the side-to-side symmetry of the flow field. Top-to-bottom symmetry also existed in the duct, but the results shown here are from measurements made only in the lower half of the duct. The results shown for the upper half of the duct are rotated up from the lower half. The results for the swirling inflow case show the strong influence swirl can have on the flow field behavior. As expected, the upstream flow field clearly reflects the swirling inflow across the entire duct cross section and particularly striking is the disappearance of the downstream vortices that had developed for the nonswirling case. Clearly, swirling inflow can have a major effect on the flow field within a transition duct. Results like those shown in this figure were obtained for a range of centerline axial Mach number, and experiments were conducted on grant to obtain unsteady flow measurements for the exact same duct geometry.

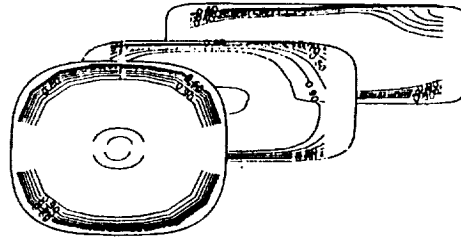


## Transition Duct—Comparison of Experimental and Computational Results

Total Pressure Contours  
Inflow Mach Number = 0.35



Experimental results

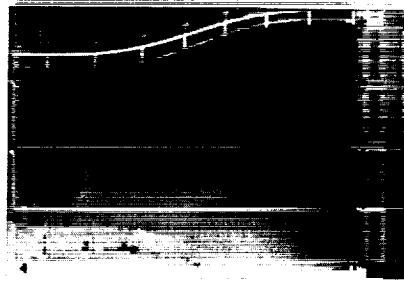


Computational results (PARC3D)

CD-91-54099

A three-dimensional, steady Navier-Stokes calculation of the flow field within the very same transition duct geometry is compared in this figure with the experimental results. The results shown are contour plots of total pressure for the swirl case with an inflow Mach number of 0.35. The basic qualitative features of the flow are well resolved by the CFD predictions. At the first station, however, the experimental results show an oval-shaped contour line above and below the centerline that does not show up in the numerical results. It is believed that this feature of the flow may be an artifact of the upstream swirl generator, or that the upstream boundary condition used by the code is not adequately modeling the true incoming conditions of the flow. Overall, the agreement between the CFD predictions and the experimental results is quite good.

## Transition Duct Heat Transfer Experiments

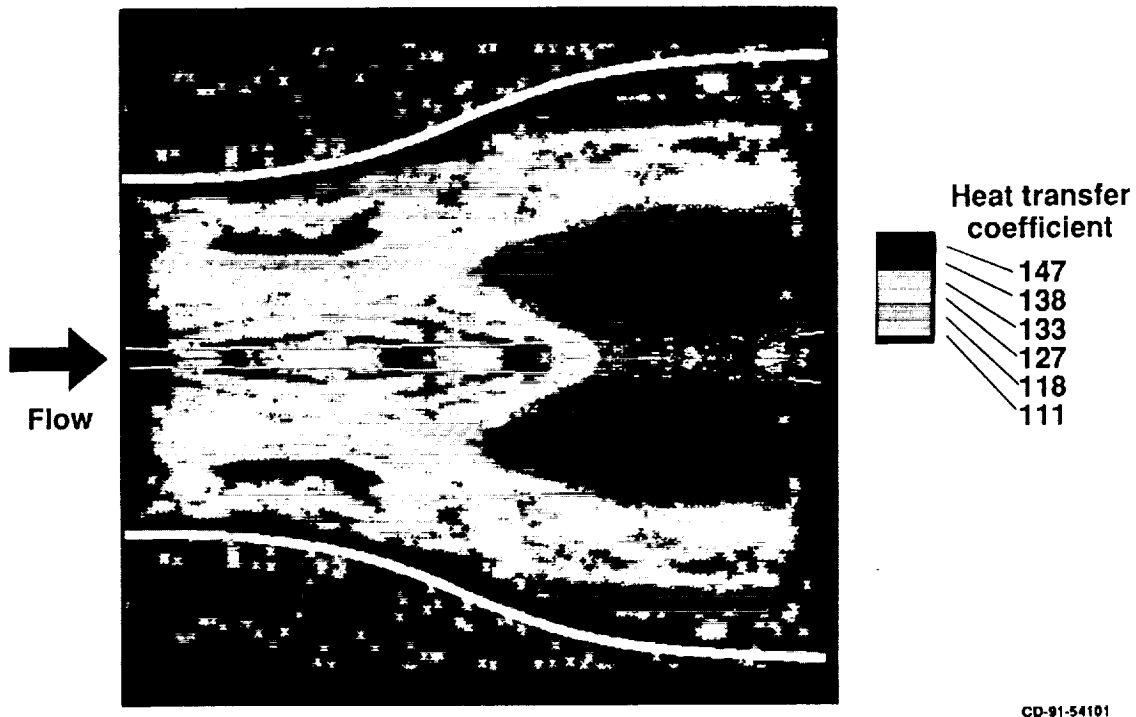


CD-91-54100

In addition to aerodynamic experiments with the highly three-dimensional flow fields of transition ducts, heat transfer experiments are also being conducted. Heat transfer is of particular concern in applications where the three-dimensional flow may result in high-temperature flow streams (i.e., engine core flow) finding their way to the transition duct (nozzle) surfaces. The figure shows the Transition Duct Heat Transfer Research Facility and the transition duct. In order to measure the surface heat transfer characteristics of the duct, the surface was coated with liquid-crystal material. The duct was then heated to an elevated uniform temperature over its entire surface. A fast-acting valve was then opened and the desired flow rate of cooler room-temperature air was quickly established within the duct. The liquid-crystal material on the duct surface then responded to this changing surface temperature, and the corresponding changes in the transient behavior of the various temperature-sensitive color bands in the liquid-crystal material were recorded. From these recorded results the value of the local heat transfer coefficient at each location on the duct surface can be determined.

## Transition Duct Heat Transfer Results

Inlet Mach Number = 0.14



A heat transfer coefficient map for the transition duct, obtained by using the transient liquid-crystal technique, is shown in this figure. The results are for a duct inlet Mach number of 0.14. The various colors on the figure represent different values of surface heat transfer coefficient. The red area (shown on the centerline at the duct exit), where the airstream impinges on the duct surface, is the region with the highest heat transfer. The green areas (shown near the corner at the duct inlet and exit), where the airstream boundary layer thickens, are the regions with the lowest heat transfer.

## **Inlets, Ducts, and Nozzles Research Program Flow Phenomena**

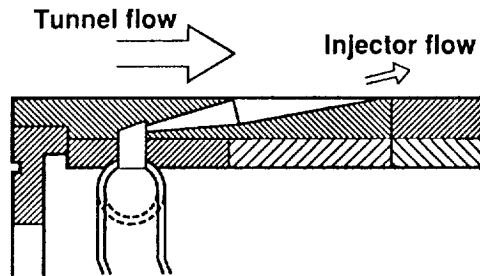
- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**
- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

CD-91-54102

Shock and high-speed-mixing flow field research will be illustrated by describing the high-speed mixing research program.

## High-Speed-Mixing Experiments

### Single Flush Nozzle



Geometry



Schlieren

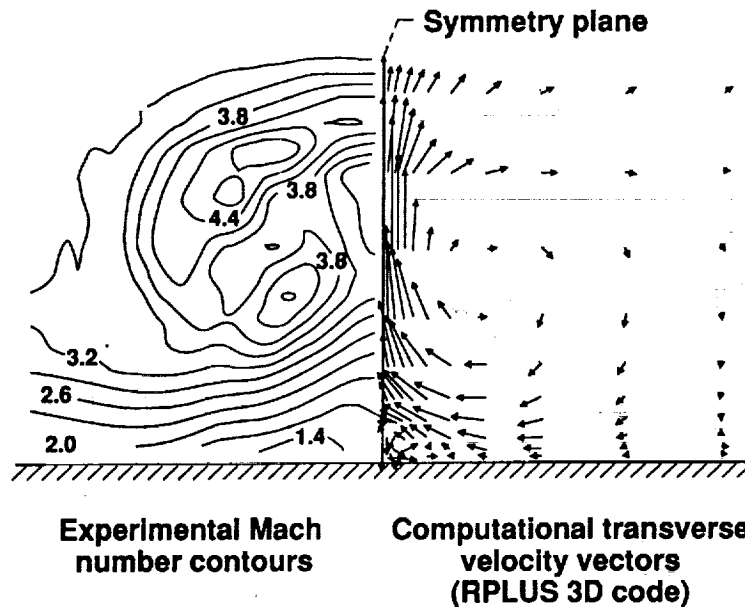
CD-91-54103

An experimental investigation is being conducted in the Lewis 1- by 1-Foot Supersonic Wind Tunnel to determine the effectiveness of various hypermixing nozzle concepts. The single flush nozzle shown in this figure was selected as a baseline model for investigating jet penetration and mixing over a range of operating conditions. Results from this nozzle configuration provide a comparison for results from more complicated multinozzle configurations. In addition, the simple geometry makes this model an ideal test case for code validation and calibration, inasmuch as complicated grid generation is avoided. The model consists of a circular underexpanded supersonic jet injected at a shallow ( $10^\circ$ ) angle into the boundary layer developing on the wind tunnel wall. The model was tested over free-stream Mach numbers from 1.6 to 3.0 with the nozzle-to-tunnel total pressure ratio varying from 6 to 30. The sample schlieren photograph illustrates some basic elements of the flow structure.

## Single Flush Nozzle Results

Free-Stream Mach Number = 3.0; Jet Mach Number = 3.5;

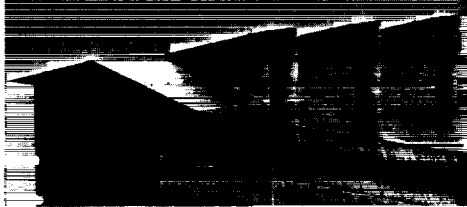
$$\frac{\text{Downstream distance}}{\text{Jet exit diameter}} = 15.0$$



Shown in this figure are both experimental and computational results for the single flush nozzle configuration. In both cases the free-stream Mach number is 3.0 and the jet exit Mach number is 3.5. The results are displayed 15 jet diameters downstream of the nozzle exit. The experimental results show a flow field that is characterized by the pressure-induced vortex pair associated with a jet in a crossflow. These vortices create the outward bulging of the contours away from the wall. In addition, the bulging of the Mach 1.4 contour near the wall, plus results from oil flow visualization photographs, hint that a second near-wall vortex pair may be present. The transverse velocity calculations shown on the right side of the figure indicate the presence of both the main vortices associated with the injection and the secondary vortices close to the wall. The agreement between the experimental and computational results appears to be quite good.

## High-Speed-Mixing Experiments

### Hypermixing Nozzle



Geometry



Schlieren

CD-91-54104

The second phase of the high-speed mixing program investigated the performance of multinozzle hypermixing configurations, one of which is illustrated in this figure along with a schlieren flow visualization photograph. The idea behind this type of nozzle concept is to utilize oblique shock wave interaction and streamwise vorticity to enhance penetration and mixing of the nozzle flow stream. The leading-edge ramp generates an oblique shock wave that then reflects from the opposing wall, crosses the jet plume, and favorably affects the jet mixing rates. In addition, the expansion troughs between the nozzles set up a pressure field that induces a strong vortex pair centered, at least initially, about the midspan of each individual nozzle. The desired effect of the vortex pair is to lift the jet up from the wall surface and thereby increase jet mixing. For the nozzle geometry shown here, the ramp angle is  $10^\circ$  and the jet injection angle is  $15^\circ$ . The free-stream Mach number ranged from 2 to 4 and the jet-to-free-stream total pressure ratio ranged from 1 to 12.

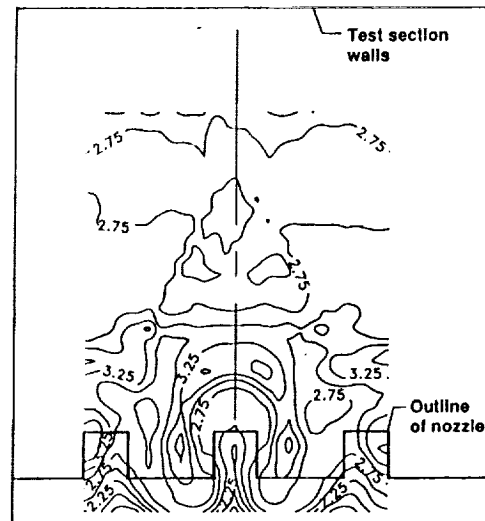
## Hypermixing Nozzle Experimental Results

Free-Stream Mach Number = 3.0;

Jet Exit Mach Number = 3.0



Oil flow pattern



Mach number contours

Downstream distance = 11.1  
Nozzle height

CD-91-54105

An oil flow visualization pattern and experimental results for the hypermixing nozzle configuration operating with a free-stream Mach number of 3.0 and a jet exit Mach number of 3.0 are shown in this figure. The oil flow pattern indicates very complex flow interactions; sidewall effects cause the near-wall flow to rapidly converge toward the wind tunnel centerline. With the exception of sagging in regions of heavy oil accumulation due to gravitational effects (the model was mounted on one of the wind tunnel vertical walls), spanwise symmetry was very good. The Mach number contours also indicate very good symmetry about the centerline of the geometry. The action of the vortex pairs (flow up) generated within the nozzle expansion troughs created the characteristic mushroom shape in the downstream contours.



## **Inlets, Ducts, and Nozzles Research Program**

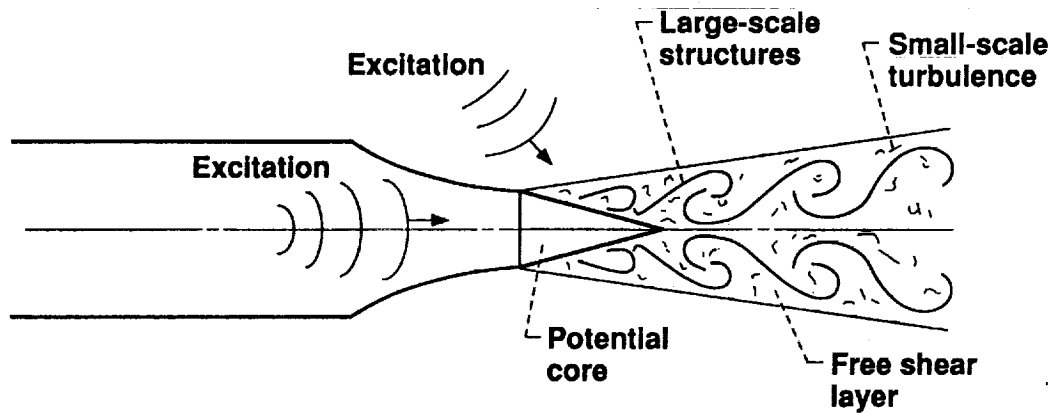
### **Flow Phenomena**

- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**
- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

CD-91-54106

Shear flow control research will be illustrated by describing the free-shear-layer-control research program.

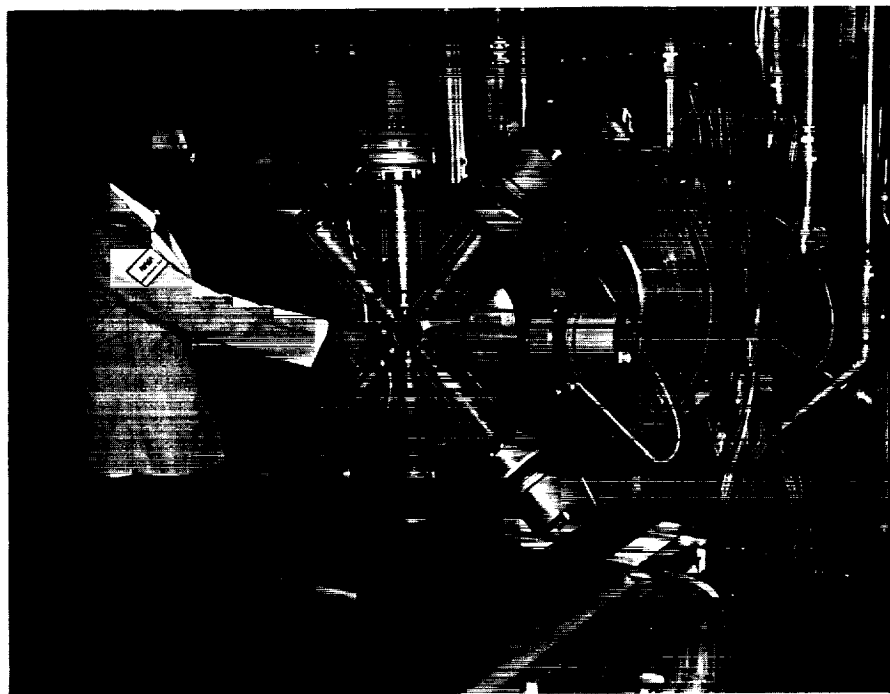
## Enhanced Free-Shear-Layer Mixing



CD-81-54107

The mixing process between a jet and the surrounding airflow can be enhanced through use of aerodynamic excitation. The naturally occurring flow structure of the jet mixing process is illustrated in this figure. By applying excitation signals at the proper frequencies, with the proper modal structure, and with the appropriate amplitude and phase, the naturally occurring large-scale, unsteady structures within the mixing layer are regularized and enhanced and lead to a more rapid mixing process between the jet flow and the surrounding air.

## Enhanced Free-Shear-Layer-Mixing Experiments

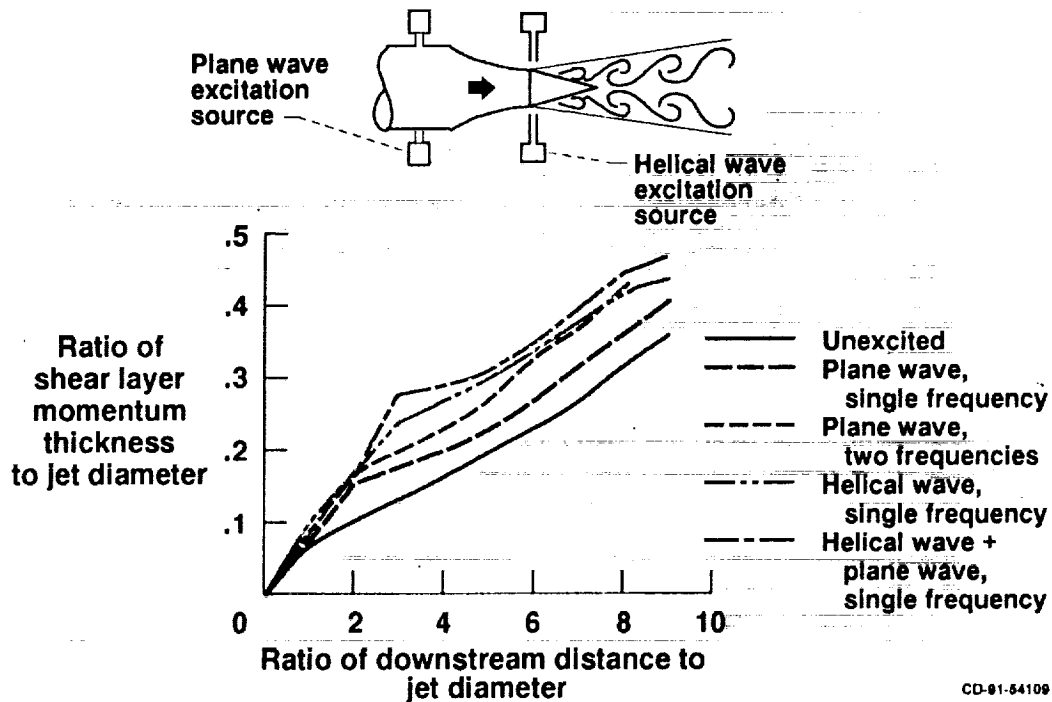


CD-91-54108

This figure shows the test facility where experiments are being conducted to assess how aerodynamic excitation affects the free-shear-layer mixing process between a circular jet and the surrounding quiescent air. The facility has been carefully designed to provide a jet exit flow with a low turbulence level and with very low internal noise levels. The controlled aerodynamic excitation signals can be provided either internal to the plenum tank or externally. The external excitation system is clearly evident here and consists of the eight acoustical drivers arranged circumferentially around the jet exit plane. These external excitation devices are used to provide helical mode excitation; the internal excitation devices are used to provide plane wave excitation. Through proper control of frequency, amplitude, phase, and modal structure, any number of aerodynamic excitation combinations can be applied to the jet free shear layer.

## Enhanced Free-Shear-Layer Mixing—Experimental Results

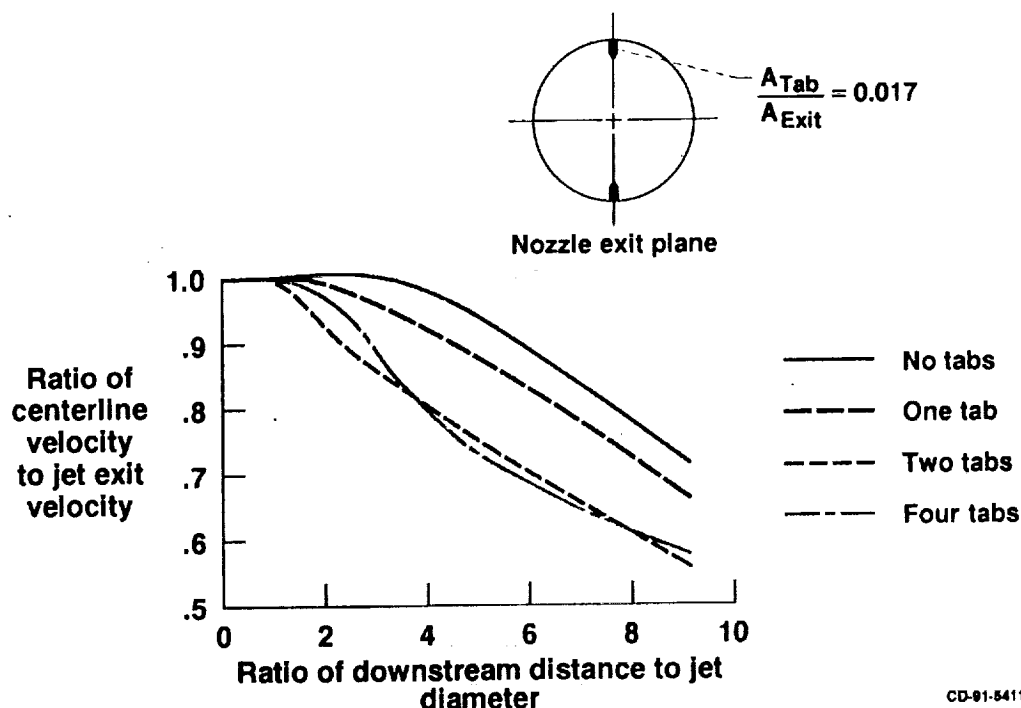
Jet Exit Mach Number = 0.2; Jet Turbulence = 0.15 Percent



The favorable influence of aerodynamic excitation on the mixing of a circular jet free shear layer is shown in this figure. The results are for various types of aerodynamic excitation in terms of the momentum thickness of the shear layer versus the downstream distance. The jet exit Mach number is 0.2 and the turbulence level within the jet at the exit is 0.15 percent. The lower curve on the figure represents the downstream growth of the jet shear layer for the natural, unexcited jet. The other four curves show the progressive effect on shear layer growth of (1) plane wave excitation applied at the preferred jet excitation frequency, (2) plane wave excitation applied at both the preferred jet excitation frequency and at half that frequency (to encourage large-scale structure "pairing"), (3) helical wave excitation applied at the preferred jet excitation frequency, and (4) both helical wave and plane wave excitation applied at the preferred frequency. The results indicate the significant influence that aerodynamic excitation can have on free-shear-layer mixing.

## Enhanced Free-Shear-Layer Mixing—Effect of Tabs

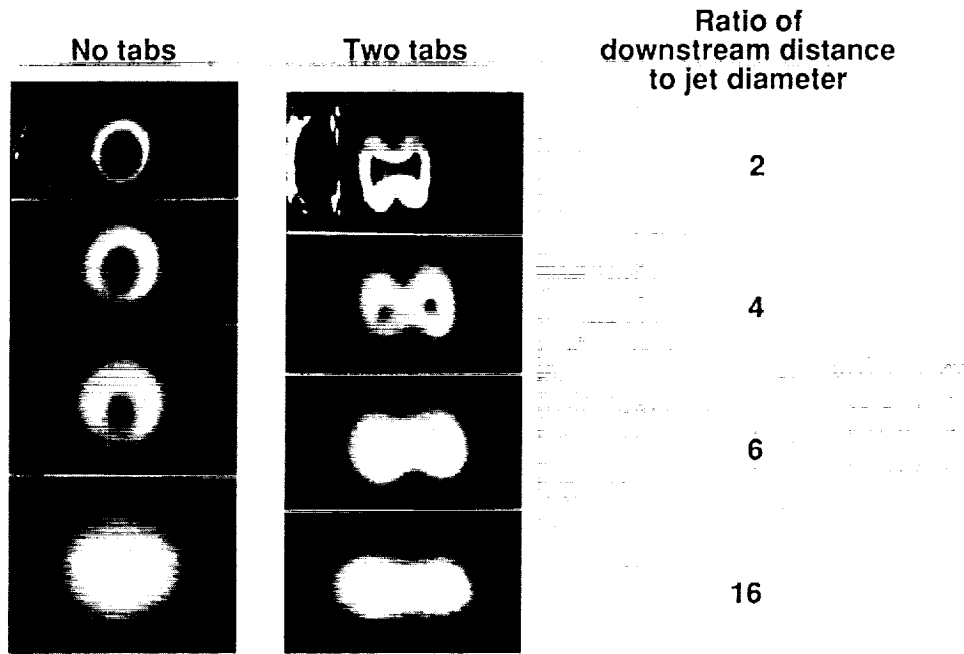
Jet Exit Mach Number = 0.34



In addition to investigating the effects of aerodynamic excitation on the mixing process of a free shear layer, the effect of tabs, or small protrusions in the flow at the exit plane of a nozzle, has also been investigated as part of the shear-flow-control research program. As indicated in the figure, these tabs are quite small, blocking only 1.7 percent of the exit flow area (per tab). How the tabs influence jet mixing is shown in this case in terms of the decay of the jet centerline velocity with downstream distance for a jet exit Mach number of 0.34. The top curve shows the natural decay of the jet centerline velocity for the jet with no tabs. The other three curves show the effect of adding one, two, and then four tabs at the nozzle exit plane. Note that significant increases in mixing (decreases in centerline velocity) occur with the introduction of one and then two tabs. The four-tab case is not too different from the two-tab case. These investigations were conducted over the Mach number range 0.34 to 1.81. Similar enhancements to jet mixing were observed at all jet exit Mach numbers.

## Enhanced Free-Shear-Layer Mixing—Effect of Tabs

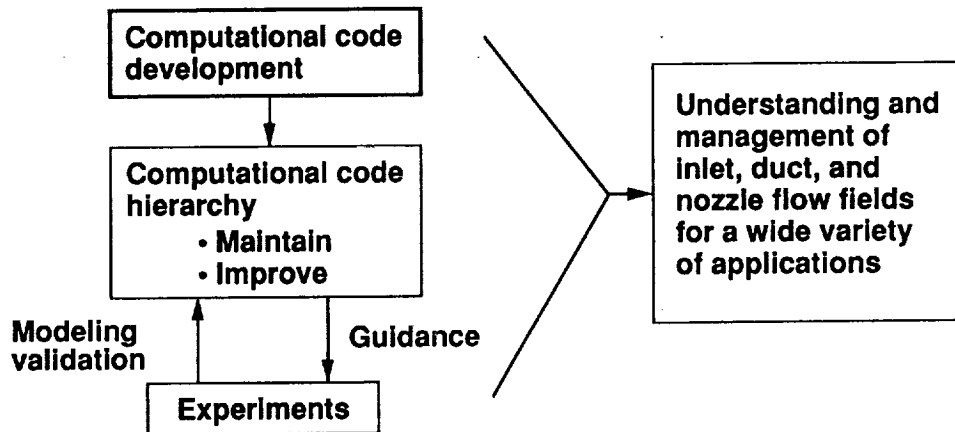
Jet Exit Mach Number = 1.63



CD-91-54111

The flow visualization results shown in this figure illustrate the changes that occur in the shape of the jet mixing layer when tabs are introduced into the nozzle exit plane. The jet exit Mach number was 1.63. Results are shown for the no-tab and the two-tab cases at four downstream locations. The images were obtained by passing a sheet of laser light through the jet, perpendicular to the jet centerline. Because of the high jet Mach number the static temperature of the jet flow was far enough below the room air dewpoint that the moisture in the room air, which had mixed in with the jet air in the shear layer, condensed and formed the images shown. As expected, for the no-tab case the mixing layer remained circular in cross section with the jet core vanishing just downstream of six jet diameters. The two-tab case, however, essentially split the jet core into two jets and this effect persisted until at least 16 diameters downstream. The jet clearly spread more in the plane perpendicular to the line containing the two tabs.

## Inlets, Ducts, and Nozzles Research Program Approach



CD-91-54112

Thus far, this description of the internal fluid mechanics research program in inlets, ducts, and nozzles has focused on showing the results of experiments and the application of computer codes to several flow phenomena of interest. Returning to this earlier figure, another element of the program involves the development of new computer codes, which are added, as necessary, to the existing hierarchy of codes, to be used for the stated objective of improving the understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications. By way of closing this presentation, one such code development effort, currently under way, will be described.

## **PROTEUS Code Development**

- **Objective**

- **Develop user-oriented, easily modified, well-documented, two- and three-dimensional, unsteady Navier-Stokes codes for aerospace propulsion applications**

- **Approach**

- **Use well-proven, state-of-the-art algorithms**
- **Use a consistent, modular code structure**
- **Emphasize documentation, code readability, and validation**

CD-91-54113

An effort is currently under way to develop two- and three-dimensional unsteady Navier-Stokes codes, called PROTEUS, for aerospace propulsion applications that include inlets, ducts, and nozzles. The emphasis in the PROTEUS effort is not algorithm development or research on numerical methods but rather the development of the code itself. The objective is to develop codes that are use oriented, easily modified, and well documented. Well-proven, state-of-the-art solution algorithms are being used. Code readability, documentation (both internal and external), and validation are being emphasized.



## **PROTEUS Validation Cases**

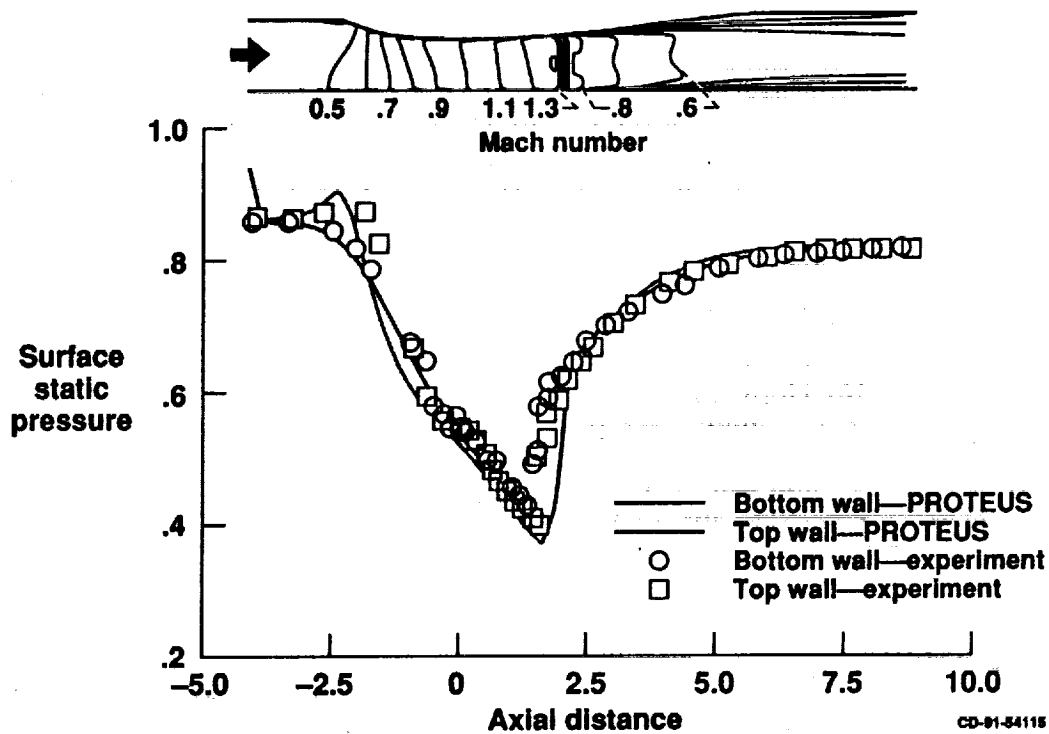
- **Flow over a flat plate**
  - Blasius solution for laminar flow
  - Laminar flow with confined separation
  - Turbulent flow with or without heat transfer
- **Exact Navier-Stokes solutions**
- **Driven cavity**
- **Development of laminar and turbulent flow in a channel and pipe**

• <b>Steady and unsteady turbulent flow in a transonic diffuser</b>
---

CD-91-54114

An extensive series of validation cases have been run, primarily using the two-dimensional planar/axisymmetric version of PROTEUS. Some of the major validation cases are listed here and include comparing PROTEUS results with exact Navier-Stokes solutions, with experimental results, and with results from other numerical calculations.

## Transonic Diffuser Validation



A typical PROTEUS validation case is shown in this figure. The geometry consists of a two-dimensional transonic diffuser operating at a maximum Mach number of about 1.3 prior to shocking down to subsonic conditions. PROTEUS results are compared with experimental data taken on both the top and bottom surfaces of the diffuser. The agreement between the PROTEUS results and the experimental data is quite good over the full length of the diffuser.

## **PROTEUS Code Status**

- **Two-dimensional version released in September 1989**
- **Three-dimensional version release scheduled for September 1991**
- **Users workshop to be held in spring of 1992**

CD-91-54118

A two-dimensional version of the PROTEUS code was released in September 1989 along with a three-volume set of documentation. The three-dimensional version of the code is scheduled for release in September 1991. A PROTEUS users workshop will be held in the spring of 1992.

